

DESIGN, CONSTRUCTION, AND TESTING OF AN OPEN-LOOP LOW-SPEED
WIND TUNNEL

MUHAMMAD ADNIN BIN MAT BAHARI

A report submitted in partial fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

ABSTRACT

An open-loop low-speed wind tunnel is one of the easiest ways to study about aerodynamics for undergraduate studies. The objectives of this project are to propose a design with detail analysis, fabrication of a small scale open-loop low-speed wind tunnel and to validate the designed wind tunnel through performance testing with the existing instrumentations available in the laboratory. The wind tunnel was designed by considering the essential parts of the wind tunnel with the proper justifications before modelled with Computer Aided Design (CAD) and then tested using the Computational Fluid Dynamics (CFD). After obtaining the desired simulation result, the designed wind tunnel was fabricated and then followed by the test models. Then the wind tunnel undergoes the performance testing for validation and calibration. For the Ahmed Body flow pattern testing, the flow behaves just like the flow pattern tested in calibrated wind tunnel. For the case study testing, a cylinder model was used and the highest flow speed is 0.4317 m/s while the slowest flow speed is 0.1401 m/s. However for the case study experiment, the result obtained is not at its best condition as there is wake flow generated around the cylinder body and further improvement is required to obtain the undoubtedly results.

ABSTRAK

Terowong angin terbuka berhalaju rendah adalah salah satu daripada cara yang termudah untuk mempelajari perkara yang berkenaan dengan aerodinamik. Objektif untuk projek ini adalah untuk menghasilkan rekaan yang teliti dan membina terowong angin terbuka serta menguji keupayaan terowong angin tersebut dengan kelengkapan yang ada di makmal. Terowong angin tersebut direka dengan mengambil kira justifikasi-justifikasi yang betul pada bahagian-bahagian yang penting sebelum dimodelkan dengan *Computer Aided Design* (CAD) dan disimulasikan dengan menggunakan *Computational Fluid Dynamics* (CFD). Setelah mendapatkan keputusan simulasi yang diinginkan, terowong angin terus dibina dan diikuti oleh model ujian. Kemudian terowong angin tersebut diuji untuk penilaian keupayaan supaya setaraf dengan terowong angin yang telah lulus penilaian. Untuk eksperimen corak arus menggunakan *Ahmed Body*, corak arus yang diperolehi menunjukkan ciri-ciri persamaan yang ketara dengan corak arus yang diperoleh melalui model yang diuji dengan terowong angin yang ideal. Untuk kajian kes, model silinder telah digunakan dan halaju arus yang paling tinggi adalah 0.4317 m/s manakala halaju arus yang paling rendah adalah 0.1401 m/s. Walaubagaimanapun, eksperimen bagi kajian kes, bacaan-bacaan yang diperolehi adalah bukan bacaan yang terbaik kerana terdapat arus tidak stabil terhasil di sekeliling model silinder dan beberapa penambahbaikan perlu dilakukan untuk mendapatkan keputusan kajian yang tidak diragui.

TABLE OF CONTENTS

	Page
SUPERVISOR’S DECLARATION	ii
STUDENT’S DECLARATION	iv
DEDICATION	v
ACKNOWLEDGEMENTS	vi
ABSTRACT	vii
ABSTRAK	viii
TABLE OF CONTENTS	viii
LIST OF FIGURES	xiii
LIST OF TABLES	xv
LIST OF SYMBOLS	xvi
LIST OF ABBREVIATIONS	xvii
 CHAPTER 1 INTRODUCTION	
1.1 Project Background	1
1.2 Problem Statement	1
1.3 Project Objective	2
1.4 Scope of study	2
 CHAPTER 2 LITERATURE REVIEW	
2.1 Introduction	3
2.2 Design of wind tunnel	3
2.2.1 Theory of wind tunnel	4
2.2.2 Bernoulli’s principle	4
2.2.3 Effect of flow and pressure on Bernoulli’s Principal	8

2.2.4 Reynold's number effect	9
2.2.5 Wind tunnel classification	9
2.2.6 Speed regime	10
2.2.7 Tunnel geometry	11
2.2.8 Working fluid	12
2.3 Fabrication of wind tunnel	13
2.3.1 Design parameters	13
2.3.2 Contracting cone	14
2.3.3 Honeycomb	15
2.3.4 Testing chamber	16
2.3.5 Diffuser	17
2.4 Testing of wind tunnel	17
2.4.1 Ahmed Body Theorem	18
2.4.2 Pressure measurement	19
2.5 Summary	20

CHAPTER 3 METHODOLOGY

3.1 Introduction	22
3.2 Flow chart of study	22
3.3 Conceptual study	24
3.4 Design method	24
3.4.1 Contracting cone design	25
3.5 Design justification	28
3.5.1 Item consideration	28
3.5.2 Contracting cone dimensions	28
3.5.3 Entrance length formula, El	28
3.5.4 Contraction ratio, c	30
3.6 Design Analysis	31

3.7	Model selection	31
3.7.1	Ahmed Body	32
3.7.2	Case study: Cylinder	32
3.8	Fabrication process	32
3.8.1	Wind tunnel	33
3.8.2	Position of model testing	35
3.8.3	Position of smoke generator outlet	36
3.9	Experimental technique	36
3.10	Simulation technique	37

CHAPTER 4 RESULTS AND DISCUSSIONS

4.1	Introduction	41
4.2	Selection of contracting cone	41
4.2.1	Contracting cone	41
4.2.2	Testing chamber	47
4.2.3	Honeycomb effect	50
4.3	Calculation of Reynolds number	51
4.3.1	Reynold's number of Ahmed Body and cylinder model	52
4.4	Ahmed Body analysis	52
4.4.1	Analysis on front part	52
4.4.2	Analysis on slant angle	53
4.5	Discussion on Ahmed Body	55
4.5.1	Smoke generator	55
4.5.2	Air gaps	55
4.5.3	Honeycomb alignment	56
4.5.4	Summary	44
		56

4.6	Cylinder Model Analysis	56
4.6.1	Introduction	56
4.6.2	Result	57
4.7	Discussion on case study	58
4.7.1	Wake region	59
 CHAPTER 5 CONCLUSION		
5.1	Conclusions	60
5.2	Recommendations	61
5.2.1	Material change	61
5.2.2	Fan change	61
5.2.3	Portability	61
5.2.4	Adjustable cone	62
 REFERENCES		63
APPENDIX		65
 Appendix A: Isometric wind tunnel		66
Appendix B: Wind tunnel full dimension		67

LIST OF FIGURES

Figure No.	Title	Page
2.1	A pitot tube system	7
2.1(a)	Subsonic wind tunnel system	10
2.1(b)	Supersonic wind tunnel system	10
2.1(c)	Hypersonic wind tunnel system	10
2.2(a)	A conceptual diagram of an open-loop wind tunnel	11
2.2(b)	A conceptual diagram of a closed-loop wind tunnel	12
2.3	Water tunnel system	13
2.4	Open-loop and closed-loop wind tunnel comparison diagram	14
2.5	Wooden contracting cone	15
2.6	Honeycomb (air flow straightener)	16
2.7	Testing chamber of a wind tunnel	16
2.8	Diffuser diagram of a wind tunnel	17
2.9	Vortex region of an Ahmed Body	19
2.10	Multi-tube manometer at Fluid Mechanics Laboratory	20
3.1	Project flow chart diagram	23
3.2	6:1 contracting cone	25
3.3	8:1 contracting cone	26
3.4	9:1 contracting cone	27
3.5	The dimension of Ahmed Body model.	32
3.6(a)	A completed wind tunnel	34
3.6(b)	A testing chamber	35
3.6(c)	A honeycomb (329 cells)	35
3.6(d)	An axial exhaust fan	35
3.6(e)	A set of a hot-wire anemometer	35

3.7(a)	A side view of the model	36
3.7(b)	A top view of the model	36
3.8	An anemometer sensor inside the test section	37
3.9	Computational domain and boundary condition	38
3.10	Grid refinement around the Ahmed body model	39
3.11	A velocity testing using Solidworks 2011 Flow Simulation	40
4.1(a)	Air flow speed simulation for 6:1 contracting cone	42
4.1(b)	Air flow pressure simulation for 6:1 contracting cone	42
4.2(a)	Graph of Air Flow Speed against Points	44
4.2(b)	Graph of Air Flow Pressure against Points	45
4.3	Pressure distribution points inside the testing chamber	47
4.4	Graph of pressure distribution against points inside the testing chamber	48
4.5	An isometric view of the designed honeycomb	50
4.6(a)	The velocity cutplot of the hexagonal honeycomb	50
4.6(b)	The Pressure cutplot of the hexagonal honeycomb	51
4.7	The position of the Ahmed Body before the experiment	53
4.8(a)	Smoke flow pattern at the slant surface and rear surface (Top view)	53
4.8(b)	Smoke flow pattern at the slant surface and rear surface (Side view)	53
4.9(a)	High drag flow structure	54
4.9(b)	Low drag flow structure	54
4.10	The graph of velocity, m/s versus the points at 2.0 m/s velocity of air flow	58

LIST OF TABLES

Table No.	Title	Page
4.1	Air flow speed and pressure distribution for 6:1, 8:1 and 9:1 cone	43
4.2	Table of pressure difference inside contracting cones	46
4.3	The air flow pressure distributions inside every contracting cone	48
4.4	Reynold's Number for Ahmed Body and Cylinder model	52
4.5	Reynold's Number and drag coefficient for cylinder	56
4.6	Velocity at $V = 2.0$ m/s	57

LIST OF SYMBOLS

Re	Reynolds Number
V, v	Fluid velocity
D	Lateral dimension (diameter)
L	Lateral dimension (length)
ν	Kinematic viscosity of fluid
μ	Dynamic viscosity of fluid
ρ	Fluid density
El	Entrance length
Le	Length of entrance
Cr	Contraction ratio
m	meter
cm	centimeter
mm	millimeter
$^{\circ}C$	Degree Celsius

LIST OF ABBREVIATIONS

CAD	Computer-Aided Design
CFD	Computational fluid dynamic
CR	Contraction ratio
3D	3 Dimensional
2D	2 Dimensional

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

The process for design, fabrication, and testing of an open-loop low-speed wind tunnel is discussed in this report. Important parameters in design and analysis of the small scale wind tunnel will also be presented in this chapter.

Commonly, the wind tunnel is widely used in laboratory research as well as for automotive and aeronautics industries. For instance, wind tunnel is used to enhance the performance of cars such as the new development of windshield or front bumper for better fuel efficiency and also to reduce the wind-noise. Moreover, another advantage of using wind-tunnels is the experiments can be performed under well controlled steady flow circumstances compared to experiments in the open environment. Before the wind tunnel can be used to carry out experiment, it must have instrument devices to measure the system for the wind tunnel. Generally, series of studies had been conducted in this project to make sure that the wind tunnel can carry out the experiment with the aid of several devices to be attached to the wind tunnel to come up with reliable results.

1.2 PROBLEM STATEMENT

Previously, there is no reliable small scale suction type have been developed in UMP. However, there is one discharge type wind tunnel that is available in the laboratory. But, the air flow generated inside the wind tunnel is interrupted by the fan's blade that caused the air flow to be unsteady flow. According to aerodynamic study, the

air properties are very essential in order to study the flow behaviour when it is in contact with a solid body.

1.3 PROJECT OBJECTIVE

The objectives of this project are:

- (i) To propose a design, fabricate and perform detail analysis of an open-loop low-speed wind tunnel.
- (ii) To validate the performance of the designed wind tunnel through the testing with the existing instrumentations available in the laboratory.

1.4 SCOPE OF STUDY

The scope of this project consists of:

- (i) Detail design and analysis about an open-loop low-speed wind tunnel. The detail design consists of the justification of every essential part of the wind tunnel such as the contracting cone, honeycomb, testing section, axial fan and also the diffuser.
- (ii) Computational Fluid Dynamic (CFD) to aid the designation of the wind tunnel. In this project, the CFD software used is Solidworks Flow Simulation 2011.
- (iii) Validation and calibration of the designed wind tunnel using Ahmed Body and cylinder model.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

In the nineteenth century, the astronauts to-be had studied the flight principles of the birds in order to design an aircraft. Then, humans start to make many kinds of flying machines and aviation structures. Carrying a big hope, a birdlike structure was repetitively constructed and tested in order to bring human to fly like birds. Unfortunately, the flying structure was failed each time it was tested. Soon after that, various approaches of aerodynamic studies were carried out to improvise the flying machine. Before the existence of wind tunnel, the whirling arm was used to study the air flow behavior on aircraft surfaces. The whirling arm allows people to study about the aerodynamic factors that should be emphasized in aircraft design by moving the aircraft part around in normal atmospheric condition. However, to enhance the result of the study, a better device needs to be designed. Until 1871, Frank Wenham, a Council Member of the Aeronautical Society of Great Britain, designed and operated the first wind tunnel using a steam engine powered blower fan. This first achievement of wind tunnel construction triggered a rapid development of aerodynamic study on aircraft model until now.

2.2 DESIGN OF WIND TUNNEL

In order to construct a reliable open-loop low-speed wind tunnel, the studies about designation of wind tunnels have to be carried out to avoid from making any mistakes while designing, fabricating and also testing the wind tunnel.

2.2.1 Theory of wind tunnel

According to Woodford (2011) in his journal, the basic idea of a wind tunnel is crude and simple. It is like a huge pipe that wraps around on itself in a circle or square with a fan in the middle. Then, switch on an axial fan and the air flow will flow through it. After that, place a little door so that a model can get in a test room in the middle of the pipe and there is a wind tunnel.

In practice, it is a bit more sophisticated than that. Instead of being uniformly shaped all the way round, the pipe is wider in some places and much narrower in others. Where the pipe is narrow, the air has to speed up to get through. The narrower the pipe, the faster it has to go. It works just like a bicycle pump, where the air speeds up when you force it out through the narrow nozzle, and like a windy valley where the wind blows much harder, focused by the hills on either side.

Aerodynamicists use wind tunnels to test models of proposed aircraft. In the tunnel, the engineer can carefully control the flow conditions which affect forces on the aircraft. By making careful measurements of the forces on the model, the engineer can predict the forces on the full scale aircraft. And by using special diagnostic techniques, the engineer can understand and improve the performance of the aircraft.

Wind tunnels are designed for a specific purpose and speed range and there is a wide variety of wind tunnel types and model instrumentation. The model to be tested in the wind tunnel is placed in the test section of the tunnel. The speed in the test section is determined by the design of the tunnel. The choice of speed range affects the design of the wind tunnel due to compressibility effects.

2.2.2 Bernoulli's principle

The relationship between the velocity and pressure effects by a moving liquid is described by the Bernoulli's principle which states that as the velocity of a fluid increases, the pressure exerted by that fluid decreases. This principle is valid for both liquid and gas. Airplanes get a part of their lift by taking advantage of Bernoulli's

principle. Race cars employ this principle to keep their rear wheels on the ground while travelling at high speeds. The Continuity Equation relates the speed of a fluid moving through a pipe to the cross sectional area of the pipe. It says that as a radius of the pipe decreases the speed of fluid flow must increase and vice-versa.

In practice, for inviscid incompressible (i.e., ignoring the effects of viscosity and assuming density is constant) follow the principle of continuity and the Bernoulli equation become useful in determining average flow properties along a streamline. Time-average velocity measurements can be made with a pitot-static tube. The operation of a pitot-static tube is based upon the Bernoulli equation which for a steady incompressible flow takes the form. The Bernoulli's principle states for a fluid flowing at a certain velocity, pressure surrounding this fluid will be decreased. The explanation of the principle can be seen through its equation, the Bernoulli equation. This equation is an approximate relation between pressure, velocity and elevation. It is valid in regions of steady, incompressible flow where net frictional forces are negligible. The Bernoulli equation can be stated as:

The continuity equation is given by Equation 2.1 as below:

$$AV = \text{Constant} \rightarrow \frac{dA}{A} = - \frac{dV}{V} \quad (2.1)$$

Equation 2.1 is the continuity equation that representing the nature law of air velocity when the air flowing through a pipe or duct. The increment of the flowing area will cause the air velocity to decrease. The decrement of air velocity will occur along the increment of the passing area. In other word, the air velocity is inversely proportional to the passing area in a pipe or duct. This equation is required while designing the wind tunnel as the equation proving that the area factor is very important in designing the contracting cone and the diffuser.

Putting Bernoulli into the continuity equation given by the Equation 2.2:

$$V_m^2 = 2 \frac{C^2}{C^2 - 1} \frac{P_{\text{settling}} - P_m}{\rho} \approx 2 \frac{\Delta p}{\rho} \quad (2.2)$$

The Bernoulli equation can be obtained or derived by applying the conservation of linear momentum principle. The key approximation in the derivation of the Bernoulli equation is that viscous effects are negligibly small compared to inertial, gravitational and pressure effects. In theory, this fluid is called inviscid fluid.

The operation of a pitot-static tube is based upon the Bernoulli equation which for a steady incompressible flow takes the form. The Bernoulli's principle states for a fluid flowing at a certain velocity, pressure surrounding this fluid will be decreased. The explanation of the principle can be seen through its equation, the Bernoulli equation. This equation is an approximate relation between pressure, velocity and elevation. It is valid in regions of steady, incompressible flow where net frictional forces are negligible. The Bernoulli equation can be stated as Equation 2.3:

$$\frac{P}{\rho g} + \frac{v^2}{2g} + z = \text{constant} \quad (2.3)$$

Where,

$$\frac{P}{\rho g} = \text{static head}$$

z = elevation

$$\frac{v^2}{2g} = \text{dynamic head}$$

From that equation, the proper manner of understanding about the flow speed of wind tunnel at each point is:

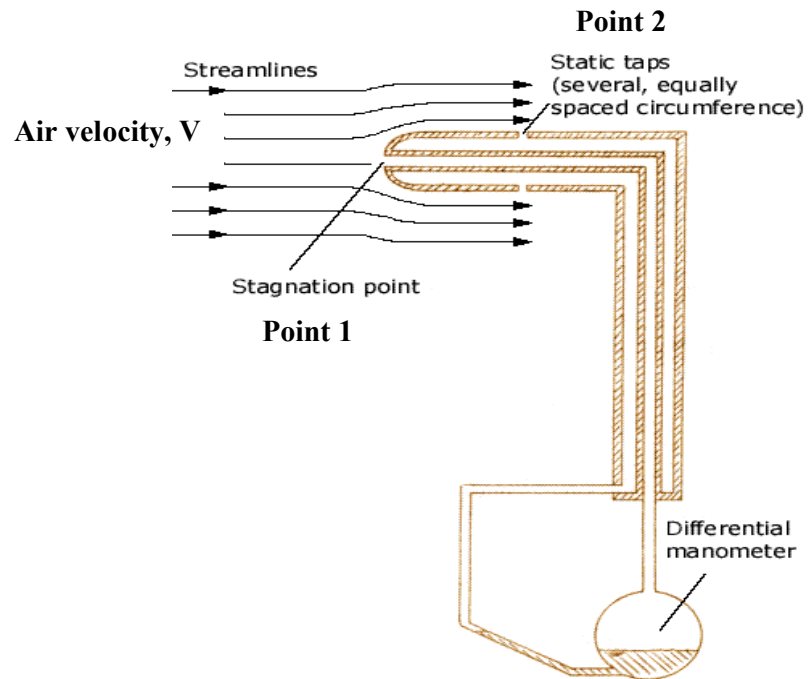


Figure 2.1: A pitot tube system

Source: Pitot, 1732

From the figure 2.1, there are Point 1 and Point 2 at in front of the pitot tube. Stagnation points exist at the surface of objects in the flow field, where the fluid is brought to rest by the object.

$$\frac{P_1}{\rho} + \frac{v_1^2}{2} + gZ_1 = \frac{P_2}{\rho} + \frac{v_2^2}{2} + gZ_2 \quad (2.4)$$

Where,

$$v_2 = 0$$

$$Z_1 = Z_2$$

$$\frac{P_2 - P_1}{\rho} + \frac{v_2^2 - v_1^2}{2} = 0 \quad (2.5)$$

From the Bernoulli's equation shows that the static pressure is highest when the velocity is zero and hence static pressure is at its maximum value at stagnation points. Hence, this static pressure is called the stagnation pressure. Point 2 is at the stagnation point where the velocity is at that point is 0. Stagnation pressure is equal to the dynamic pressure plus static pressure. Hence, the total pressure is equal to dynamic 8 pressures plus static pressure so, in incompressible flows, stagnation pressure is equal to total pressure.

The pressure at Point1 and Point 2 can be measure with using hydrostatics equation, which shown in Equation (2.6.a) and (2.6.b):

$$P_1 = P_{atm} + gh ; \quad (2.6.a)$$

$$P_2 = P_{atm} + gh , \quad (2.6.b)$$

The Bernoulli equation could obtained or derived the by applying the conservation of linear momentum principle. The key approximation in the derivation of the Bernoulli equation is that viscous effects are negligibly small compared to inertial, gravitational and pressure effects. In theory, this fluid is called in viscid fluid.

2.2.3 Effect of flow and pressure on Bernoulli's Principal

Factor that occur the concept of Bernoulli's Principal is when the speed of a liquid or gas is low, the pressure is high, and where the speed of a liquid or gas is high, the pressure is low. Then, the speed a moving fluid or gas has an effect on the pressure exerted by the fluid decreases. As the speed of a moving liquid or gas increase, the pressure within that fluid decreases. When flow speed is at high velocity, liquid or gas trade their kinetic energy for pressure. Last factor for the drag characteristic is the faster velocities exert less pressure than slow moving velocities. (Kenneth Leet, Chia-Ming Uang. 2004).

2.2.4 Reynolds number effect

Osborne Reynolds first introduced the dimensionless constant that bears his name in his 1883, in a paper he published in the Philosophical Transactions of the Royal Society. The paper, “An Experimental Investigation Of The Circumstances Which Determine Whether Motion Of Water Shall Be Direct Or Sinuous And Of The Law Of Resistance In Parallel Channels”, detailed the findings of his experimental work.

The vortex shedding from a bluff body is a function of the Reynolds Number (Re). The flow characteristics of wind passing across bluff body are depend on how the magnitude of inertial to viscous within the flow.

The Reynolds Number is defined as:

$$= \frac{\rho V L}{\mu} \quad \text{or} \quad = \frac{\rho V L}{\mu} \quad (2.7)$$

Where,

V = the wind velocity, D or L is the lateral dimension of the body,

ν = kinematics viscosity of air,

μ = dynamic viscosity of air,

ρ = density of air

2.2.5 Wind tunnel classification

There are great deals of way to classify the wind tunnel. Several types of wind tunnel have been invented for various functions toward the study of aerodynamic. The variously types of wind tunnel are actually the evolution of the subsonic wind tunnel. Then, the existence of other wind tunnel starts to take place in aerodynamic research. Basically, the wind tunnel can be classified by its speed regime, tunnel geometry and also the working fluid in the tunnel.

2.2.6 Speed Regime

Wind tunnels are often denoted by the speed in the test section relative to the speed of sound. The ratio of the air speed to the speed of sound is called the Mach number. Tunnels are classified as subsonic ($M < 0.8$), transonic ($0.8 < M < 1.2$), supersonic ($1.2 < M < 5.0$), or hypersonic ($M > 5.0$). The distinction of Mach number is caused by the relative importance of compressibility effects. For subsonic flows, the effects of compressibility may be neglected; for transonic and supersonic flows, compressibility effects must be considered. For hypersonic flows, we must make additional considerations for the chemical state of the gas. The scaling effects of the Mach number can be theoretically derived from the conservation of momentum of the air in the tunnel. Compressibility affects the design of the test section of a wind tunnel: for subsonic tunnels, the test section has the smallest cross-sectional area of the tunnel; for supersonic tunnels, the throat of the nozzle has the smallest area and the test section area is chosen to achieve a desired Mach number in the test section. The wind tunnels are shown in Figure 2.1(a) through 2.1(c).

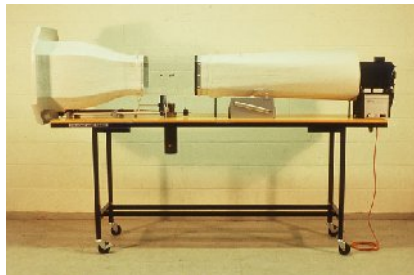


Figure 2.1(a): A subsonic wind tunnel. **Figure 2.1(b):** Supersonic wind tunnel.



Figure 2.1(c) : A closed loop hypersonic wind tunnel.

Source: Benson, 2009

2.2.7 Tunnel Geometry

Tunnel Geometry is one of the big factors to classify the wind tunnel. Actually the wind tunnel geometry can be separated in to two sections, an open-loop and a closed-loop wind tunnel.

For an open-loop wind tunnel as shown in Figure 2.2(a), the air exhausted from the diffuser did not used back into the next cycle of the flow generation, instead, the air sucked from the contracting cone is the fresh air from the surrounding air that does not under goes any compression or heating. Tom Benson (2009) states that, “A wind tunnel that is open on both ends and draws air from the room into the test section is called an open return tunnel”. The advantages of the open-loop wind tunnel is the reduction in the cost of construction as there is no heat exchanger needed for the next cycle of flow generation. In addition, the materials used are lesser as there is no ducts construction needed to connect from the tunnel’s diffuser to its contracting cone. As for the closed-loop wind tunnel as shown in Figure 2.2(b), the geometry is very different from the open-loop wind tunnel. Closed-loop or as known as closed return wind tunnel requires a duct construction to connect the diffuser back to the entrance of the contracting cone. Heat exchanger is needed in this tunnel construction in order to reduce the air temperature to be used back in the next cycle of flow generation after the air temperature is slightly rose after being compressed by the fan.

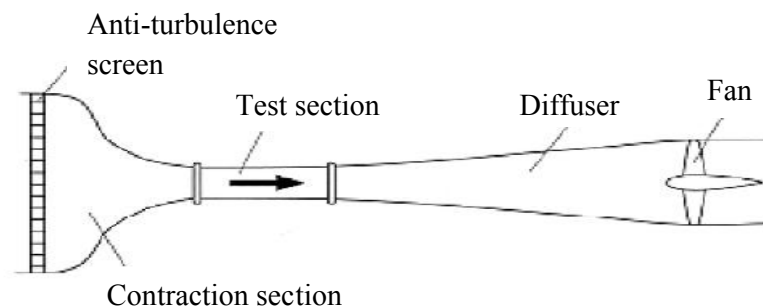


Figure 2.2(a) : A conceptual diagram of an open-loop wind tunnel.

Source: Kasravi, 2004

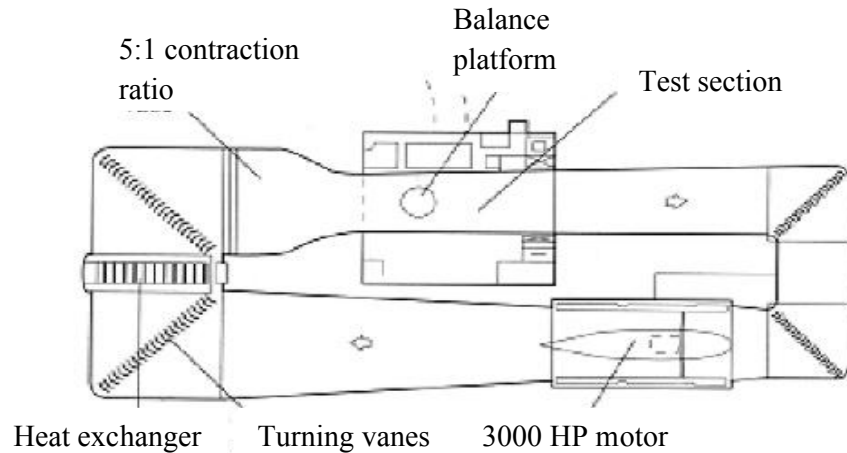


Figure 2.2(b) : A conceptual diagram of a closed-loop wind tunnel.

Source: Kasravi, 2004

2.2.8 Working Fluid

According to Tom Benson (2009), the wind tunnels can be designated by the type of fluid that is used in the tunnel. For lower speed aircraft wind tunnel testing, air moves through the tunnel. To visualize shock waves for high speed aircraft, or to study the flow around submarines or boats, water is used as the working fluid. A water tunnel is shown in Figure 2.3. In some hypersonic facilities, nitrogen or helium has been used as the working fluid. Similarly, cryogenic nitrogen has been used for high Reynold's number testing of transonic flows.

There are several wind tunnels around the world that are used to study ice build-up on aircraft parts. These icing tunnels include refrigeration devices to cool the air in the tunnel and water spray devices to provide liquid droplets in the test section.